Computers and Electronics in Agriculture 109 (2014) 200–211

Contents lists available at ScienceDirect

Computers and Electronics in Agriculture

journal homepage: www.elsevier.com/locate/compag

Numerical modelling of airflow and gas dispersion in the pit headspace via slatted floor: Comparison of two modelling approaches

Chao Zong, Guoqiang Zhang [*]

Department of Engineering, Aarhus University, Blichers Allé 20, P.O. Box 50, 8830 Tjele, Denmark

ARTICLE INFO

Article history:
Received 2 April 2014
Received in revised form 1 September 2014
Accepted 14 October 2014

Keywords:
CFD
Slatted floor
Porous media
Pollutant transport

ABSTRACT

The slatted floor system is popular in pig and cattle housing. Ammonia and odour are mostly emitted from the slurry pit under the slatted floor. In order to develop solutions to reduce this part of emissions, a better understanding of air distribution and pollutant transportation mechanisms is required. Computational fluid dynamics (CFD) is a useful technique to investigate the air motion, and transport of pollutants between room and pit headspaces via the slatted floor. However, there is a practical issue related to modelling the thousands of small slot openings in the real livestock building for CFD simulation. It is unrealistic to simulate the slatted floor with geometry details due to the large grid number and the limited computer capacity. In this study, a simplification model using porous media to represent a scaled slatted floor was developed. To assess the feasibility of this simplification, the proposed porous media model (SP) was compared with the direct geometry model (SD) and experimental data. The results showed that the porous media model was able to estimate the air velocities but not the turbulent kinetic energy. Both models predicted rotating flows under the slatted floor. A clear vertical air motion above the slatted floor was found for SP results but no such trend for SD results. The mechanism of the pollutant transportation, including the process of pollutant escaping from the pit and retention time of pollutant inside the pit headspace, was found to be inconsistent for SD and SP models. For SD, the dominant removal mechanism of transporting pollutants from the headspace to the free stream was mean flow transportation whereas it was turbulent flow transportation in SP. Higher emission rate and shorter retention time of pollutant in the headspace was obtained by using SP compared to SD. In general, though the porous media approach cannot reveal the pollutant transport mechanism, it can predict the velocity magnitude. In addition, it was found that the orientation of slats to stream flow direction plays an important role on airflow pattern and pollutant distribution inside the pit headspace.

1. Introduction

The slatted floor system is a type of floor with small slot openings which is quite popular being applied in the livestock industry. In a livestock building with a slatted floor system, pollutants like ammonia and odours are mostly emitted from the zone near the slatted floor, either the floor surface or the slurry pit under the floor (Zong et al., 2014a, b). Airflow patterns in the pit headspace and air exchange between pit and room space can significantly affect the ammonia dispersion which will further affect indoor air quality and emissions from the building (Morsing et al., 2008). Therefore, a better understanding of the airflow characteristics and mass transportation mechanisms in the pit headspace is highly desired.

A number of experimental and numerical studies have been performed on the flow and transport of pollutants in livestock buildings (Norton et al., 2009, 2010) but have been limited to the space above the slatted floor. Detailed knowledge of the characteristics of airflow and mass transport under the slatted floor is still missing, although this is the key part to predict the gas emission from the slurry pit.

As described by Wu et al. (2013), the investigated pit headspace was kind of a cubic cavity. The flow in such a cavity was featured with separation and known to be difficult to model. Other than the cavity flow in the pit headspace, similar research can be found in the area of modelling airflow in street canyons (Vardoulakis et al., 2003). The prediction of airflow and pollutant dispersion within street canyons were commonly calculated using Reynolds-averaged Navier–Stokes (RANS) models. Among those turbulence models, the standard k–ε model is the most applied and has been proved to be an accurate model for the prediction (Baik and

---

[*] Corresponding author. Tel.: +45 8715 7735.
E-mail address: GuoQiang.Zhang@eng.au.dk (G. Zhang).

http://dx.doi.org/10.1016/j.compag.2014.10.015

Kim, 2002; Johnson and Hunter, 1998; Kim and Baik, 2003; Neofytou et al., 2006; Sagrado et al., 2002).

The cavity flow around a pit headspace is much more complicated than the above-mentioned investigations of street canyons due to the involvement of the slatted floor. Up until now, very few studies on modelling pit headspace are available in the literature. Wu et al. (2012b) applied different RANS models to study the airflow characteristics under slatted floor in a 1:2 pit model of a cattle building in which the slatted floor was simulated in geometrical details. However, in a full scale livestock building, modelling slatted floor directly in geometrical details is unpractical due to the large grid number and the computer capacity (Bjerg et al., 2008b; Wu et al., 2013). The slot width in a real livestock building is up to 0.02 m while the building dimensions can be several thousand times larger. The big size difference between slot width and building dimensions including the ventilation openings prevents a direct modelling of the geometrical details for a full scale livestock building. Porous media was thus introduced to tackle this limitation in modelling slatted floor (Bjerg et al., 2008a, b; Sun et al., 2004). Up to date, the uncertainties of using porous media to simulate the slatted floor above the pit headspace have not been well documented, especially comparing with measured data. In the study of Wu et al. (2013), comparison of modelling slatted floor by using either geometrical details or as porous media was conducted, and results showed that the method of simulating slatted floor as porous media generally performed well. However, only the case in which the slats were oriented parallel to the flow direction was investigated (Wu et al., 2013). As we know, the direction of flow above the slatted floor can be different on the basis of the design of the building and air supply. For mechanically ventilated swine buildings with side wall air inlet, the dominant return airflow near the floor surface often has a direction perpendicular to the slat orientation (Zong et al., 2014a, b). An investigation of the case with the slats orientated perpendicular to the flow direction is necessary.

This study extends the investigation of airflow characteristics and ammonia dispersion around a pit headspace from a pilot study (Wu et al., 2013). The main purpose of this work is to assess the feasibility of modelling slatted floor as porous media in modelling airflow and pollutant dispersion in the pit headspace when the slats are oriented perpendicular to the flow direction. The numerical results are compared with the experimental results.

2. Materials and methods

2.1. Experimental setup

2.1.1. Wind tunnel and scaled pit model

The experiments were conducted in a wind tunnel at Air Physics Lab, Aarhus University, Denmark. Fig. 1 shows the 3.67-m long wind tunnel configuration. The wind tunnel was made of polystyrene sheets and contained a 0.8 m long transparent piece of glass to enable velocity and turbulence intensity measurements using a Laser Doppler Anemometer (LDA). A fan (Type CK125 C CBU, Lindab A/S, Denmark) was connected at the tunnel outlet to drive the air motion through the tunnel. Airflow went into the tunnel via a 0.17-m thick smooth surface contraction section fitted around the edges of the 0.35 (H) × 0.35 (W) m^2 wind tunnel cross section. Small neutrally buoyant particles made by the smoke generator (Z-series II, Antari Ltd., Taiwan) were injected into the wind tunnel inlet opening as the seeding for LDA to measure velocities.

A 1:8 scale pit model with a transparent front panel was constructed in the working section underneath the wind tunnel. The size of the scale model was 0.35 (L_p) × 0.35 (W_p) × 0.09 (H_p) m (Fig. 1b). The top of the pit model was covered by a slatted floor consisting of 17 slats. The slatted floor's upper surface was at the same level with the tunnel floor surface. The slatted floor used in this study had an opening ratio of 23.38%. The dimensions of the slat are shown in Fig. 1b. It should be mentioned that the experimental setup using the pit model and wind tunnel was primarily used for the comparison of the two numerical approaches and not intend to predict the airflow dynamic characteristics in a full scale condition.

2.1.2. Air velocity and turbulence measurement

In this investigation, air velocities and turbulence intensities were measured by a 2-dimensional Laser Doppler Anemometer (FlowExplorer System, DANTEC Dynamics A/S, Skovlunde, Denmark). Two pairs of laser beams radiated from the transmitting/receiving optics could measure the velocity horizontally and vertically. The measurement distance from the lens was 285 mm.

Air velocity and turbulence intensity profile measurements were taken at 14 different vertical heights in the pit headspace (0.005, 0.010, 0.015, 0.020, 0.025, 0.030, 0.035, 0.040, 0.045, 0.050, 0.053, 0.058, 0.065, 0.070, and 0.073 m) and at five lines L1–L5 in the X–Y plane. In addition, air velocities at two lines (R1 and R2) with eight vertical heights (0.0925, 0.095, 0.1, 0.11, 0.115, 0.145, 0.175, and 0.265 m) above the pit area in the wind tunnel space were also recorded. All the measurement positions are in the middle plane of the wind tunnel (0.175 m away from the glass window). The distribution of all measurement positions is shown in Fig. 1b. Data acquisition period at each spatial position was 600 s.

2.2. Description of numerical model

In steady state Reynolds-averaged Navier–Stokes (RANS) modelling, the instantaneous quantity is decomposed into its time-averaged and fluctuating components. The RANS equations for incompressible Newtonian fluids are the following:

$$\frac{\partial \rho \overline{u_i}}{\partial x_i} = 0 \tag{1}$$

and

$$\overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \mu\frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} - \overline{u_j'\frac{\partial u_i'}{\partial x_j}} \tag{2}$$

where $\overline{u_i}$ and $\overline{u_i'}$ are the mean and fluctuating terms of the velocity component u_i in the x_i-direction, respectively. $\overline{p}$ is the mean pressure, ρ is the air density and μ is the viscosity.

In the present study, the standard k–ε model is employed. The standard k–ε model is based on two additional model transport equations for the kinetic energy (k) and its dissipation rate (ε). Commercial software Fluent 12.0 (ANSYS, Inc., USA) was used to solve those equations. The second order upwind spatial discretization scheme was chosen for momentum, turbulent kinetic energy and turbulent dissipation rate. Standard and SIMPLE methods were employed for pressure and pressure–velocity coupling, respectively.

2.3. Computational domain and boundary conditions

Fig. 2 shows the computational domain with boundary conditions. The domain was discretized using hexahedral elements. It was observed in the experiment that the velocity gradient was zero at 0.5H height of the wind tunnel. Therefore, the half-height of the wind tunnel space was used in the CFD model, and the height of the domain was 0.265 m (0.5H + 1H_p). The free surface layer for the upstream and downstream of the pit model was extended to 3H_p and 5H_p, respectively. The measured air speed at 0.5H height